Brodric Young
ECEN 350
LTspice Clipper and Clamper Lab

ECEN 350 – LTspice Clipper and Clamper Lab (50 points)
(jas, LTspice Clipper and Clamper Lab.docx, 12/28/2024)

Note: This is a CAD lab to be done individually, rather than in teams, although please help each other out if/when opportunities arise, while still avoiding plagiarism. Submit an electronic version of a lab report to receive credit for doing this lab. The goal of your **lab report is to provide sufficient documentation so that the lab can be repeated if necessary**. Therefore, simply add to this document to arrive at your lab report, as all of the explanatory text, procedures, and Discussion and Conclusion questions contained in this document are required for a complete lab report. So, for your lab report, **add a cover page, your results, and your answers to the Discussion and Conclusions questions to the existing lab document.** Your answers to the **Discussion and Conclusions** questions are to **be uniquely yours** and not a copy of someone else's answers to these questions. Your cover page is to include class, lab title, and author. A grading rubric for this lab is included at the end of this document. The rubric does not need to be included in your lab report.

Purpose: The purpose of this lab is to better understand diode clipper and clamper circuits.

Procedure:

Part 1 – Parallel Clipper Circuit

1. Connect up the parallel clipper circuit shown below in **Figure 1** in LTspice. The sine wave voltage source symbol shown in **Figure 1** can be found in the [Misc], library as a **signal** component, although the standard voltage source also works in the simulation. Hovering the cursor over this voltage source opens an **Independent Voltage Source** pane in which the source can be configured to produce a sine wave with parameters. The transient analysis simulation command, i.e., **.tran**, can be included by means of the **Edit** pull-down menu followed by the choice SPICE Analysis. Add the net name Vin to the node between Vin and R1 for plotting purposes. There is now an older and newer version of LTspice, with some nomenclature differences between the two versions. The older version includes a **.op** for Operating Point icon along the top toolbar, whereas the newer version uses **.t** for the same function, which opens a **Spice Directive** window. A **Spice Directive** window can also be opened by choosing **Spice Directive** in the **Edit** pull-down menu in the upper left-hand corner of the LTspice window. A **Spice Directive** window is where simulation commands and device model information for parts not found in the LTspice library are entered for inclusion on an LTspice schematic. For the 6.2 V 1N5234 zener diode, first place a zener diode from the main LTspice library, referred to as **zener**. LTspice does not have a 1N5234 zener diode model, so an external model needs to be included in your schematic. First change the Zener diode type from D to D1N5234. Next open a **Spice Directive** window and paste the following model information for the 6.2 V 1N5234 zener diode:

```
.model D1N5234  D(Is=1.536f Rs=1.687 Ikf=0 N=1 Xti=3 Eg=1.11
+Cjo=130p M=.5259 Vj=.75 Fc=.5 Isr=1.719n Nr=2 Bv=6.2 Ibv=1.9685
```

+Nbv=.28384 Ibvl=7.0094e-7 Nbvl=.29418 Tbv1=443.55u)

2. In the **Edit** pull-down menu, add your name to your schematic as follows: **Edit → Aa Text.**
3. When completed with your schematic, replace the schematic shown in **Figure 1** below with your version, including your name. (9 points.)

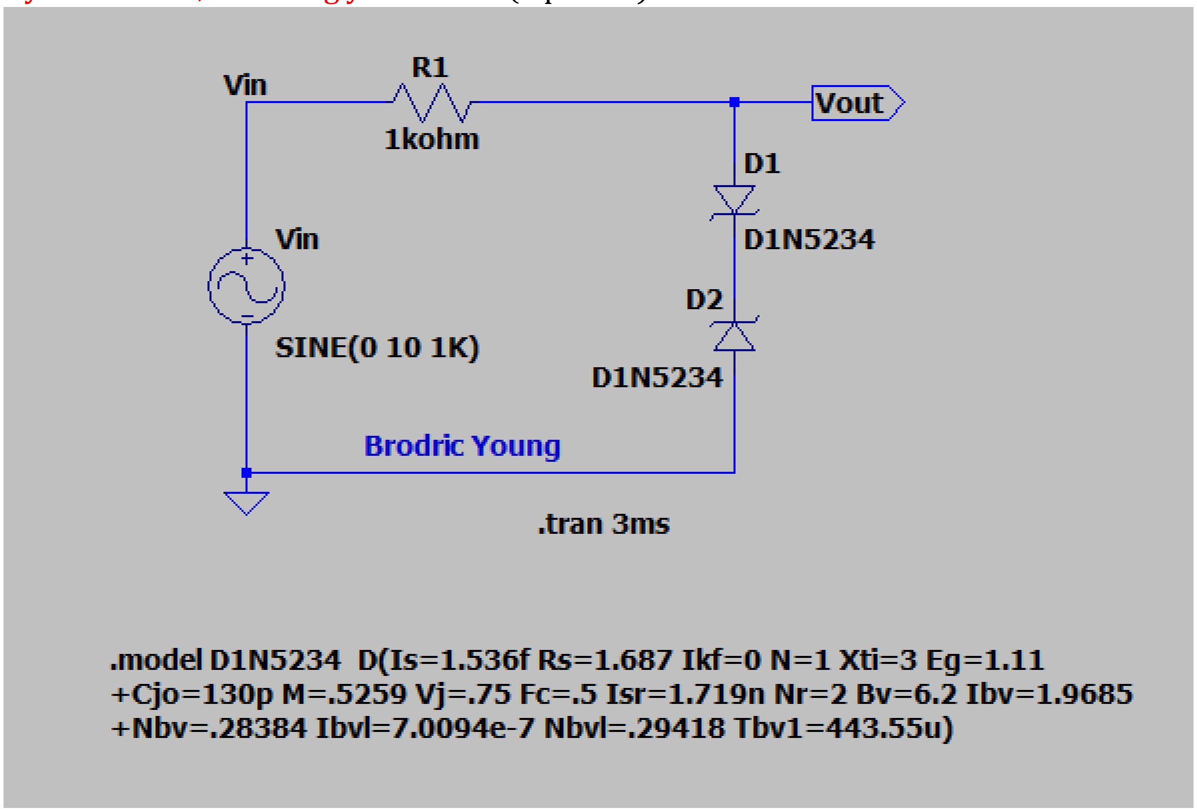

Figure 1: Parallel Clipper Circuit used for **Part 1** of the Clipper and Clamper Lab.

LTspice uses a dark background for the default plot pane, which makes it hard to visibly see some trace colors such as dark blue. The **Tools → Control Panel → Waveforms → Color Scheme → Background** allows you to modify the background color with Red = Green = Blue = 255 resulting in a white background. This is an optional change that you may find improves waveform viewing. If you make this change, it is also recommended to navigate to **Tools → Control Panel → Waveforms** and change the **Data Trace** and **Cursor width** from 2 to 3, along with using a Bold Font for the waveform labels so that the traces, cursors are waveform labels are more easily visualized.

4. After running the transient analysis, waveforms need to be chosen to display in the plot pane. This can be done by going to **Plot Settings → Add Trace.** Choose to display **V(vin)**, **V(vout)** and **I(R1)**. Note: Resistors are 2-terminal devices, with resistors placed on schematics in LTspice have the terminals defined as pin 1 and pin 2. When placing a resistor in LTspice, the resistor first appears in a vertical orientation with pin 1 being the top terminal. Spice, and therefore LTspice, have a defined resistor current polarity

as follows: Conventional current flowing into pin 1 and out of pin 2 is defined as positive, whereas the conventional current flowing into pin 2 and out of pin 1 defined as negative. Circuit voltages determine whether conventional current flows into or out of pin 1, whereas the polarity assigned to the current flowing through a resistor follows the previously defined convention. So, a negative current flow through a resistor simply means conventional current is flowing out of pin 1. If a negative resistor current is confusing or bothersome in an LTspice circuit, the offending resistor can simply be rotated by 180 ˚ so that conventional current flows into pin 1. While it may be helpful to have pin 1 indicators on resistors in LTspice, that is not the case. Rather one doesn't worry much about resistor current polarity, unless it is confusing, and then the solution is to simply rotate the resistor 180 ˚ in the schematic, or for this lab to plot **-I(R1)** instead of **I(R1)**.

5. Next the y-axis plot limits are to be adjusted to span from 12 V to -12 V with 2 V ticks. For this axis adjustment, go to **Plot Settings → Manual Limits**, which opens a **Plot Limits** pane in which you can adjust the left vertical axis limits as 12 V for the **Top** and -12 V for the **Bottom**, letting LTspice choose the **Tick** value.
6. Add both cursors to **V(vout)** by right-clicking the **V(vout)** label and under "Attached Cursor" selecting "1st & 2nd". Move the cursors to the maximum and minimum waveform values, as indicated in the figure below. Then drag the cursor pane onto your plot so as to document your maximum and minimum waveform values as illustrated in the figure below.
7. Annotate your plot pane with your name by means of the **Plot Settings** → **Notes and Annotations** → **Place Text** option available by accessing the Plot Settings pulldown menu above the toolbar.

8. Replace the resulting screen capture in **Figure 2** below with your version, leaving in the figure number and caption. (9 Points.)

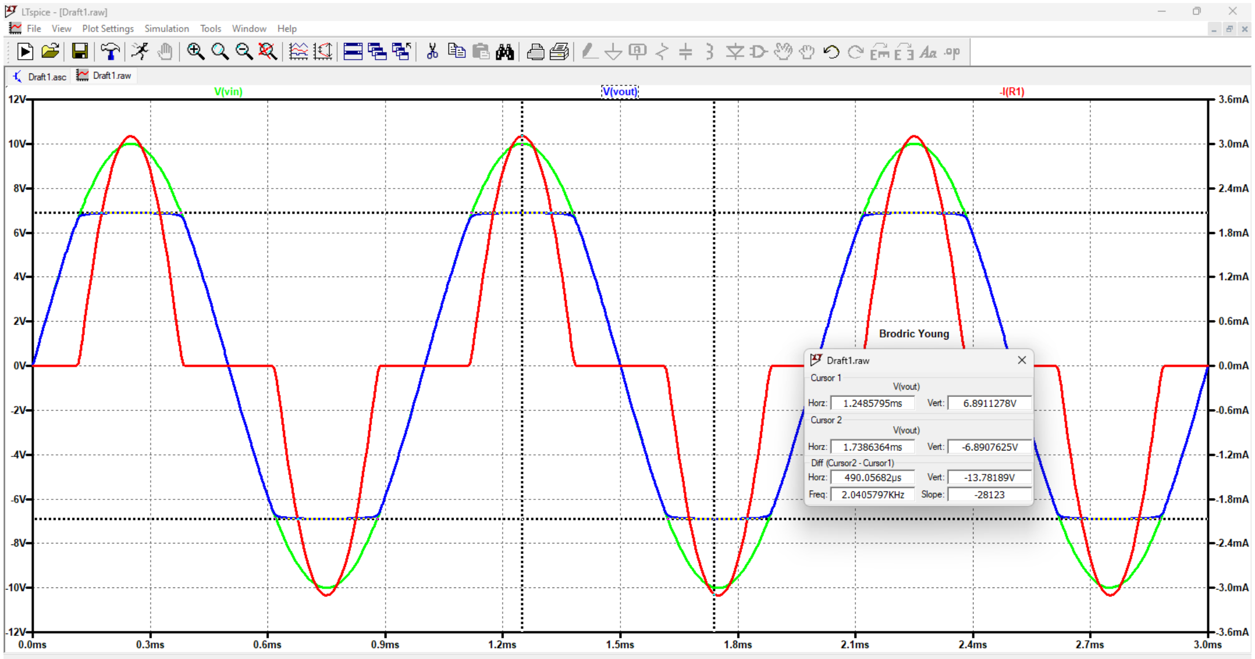

Figure 2: Input and Output Waveforms of the Parallel Clipper Circuit used for **Part 1** of the Clipper and Clamper Lab.

Part 2 – Positive Clamper Circuit

1. Connect up the parallel clipper circuit shown below in **Figure 3** in LTspice. Hovering the cursor over this voltage source opens an **Independent Voltage Source** pane. You can also right click the voltage source and select "Advanced" to find the pane. Configure the voltage source to produce a 2.5 V to -2.5 V square wave with 1 µs delay, 10 µs rise and fall times, 480 µs on time and a 1 ms period, and 20 Cycles. The transient analysis simulation command **.tran** can be included by means of the **Edit** pull-down menu followed by the choice SPICE Analysis. Add the net name Vin to the node between Vin and C1 for plotting purposes. For the 1N4148 diode, first place a generic diode, then hover the cursor over the diode and right click to open the Diode pane. Select **Pick New Diode** in the diode pane in with the 1N4148 diode can be found among the **silicon** diode types from On Semiconductor. The diode type can also be changed simply by editing the value from D to 1N4148 on the schematic.
2. In the **Edit** pull-down menu, add your name to your schematic as follows: **Edit → Aa Text.**
3. When completed with your schematic, replace the schematic shown in **Figure 2** below with your version, including your name. (7 points.)

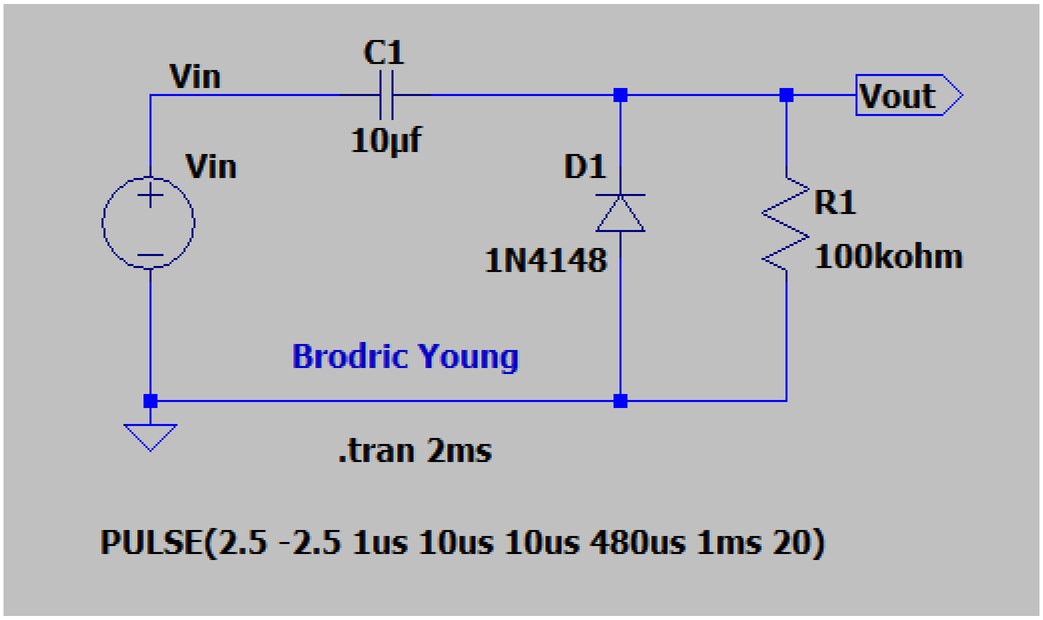

Figure 3: Positive Clamper Circuit used for **Part 2** of the Clipper and Clamper Lab.

4. After running the transient analysis, waveforms need to be chosen to display in the plot pane. Choose to display **V(vin)**, and **V(vout).**
5. Add both cursors to **V(vout)** and move the cursors to the maximum and minimum waveform values, as indicated in the figure below. Then drag the cursor pane onto your plot so as to document your maximum and minimum waveform values as illustrated in the figure below.
6. Annotate your plot pane with your name by means of the **Plot Settings** → **Notes and Annotations** → **Place Text** option available by accessing the Plot Settings pulldown menu above the toolbar.
7. Replace the resulting screen capture in **Figure 4** below with your version, leaving in the figure number and caption. (7 Points.)

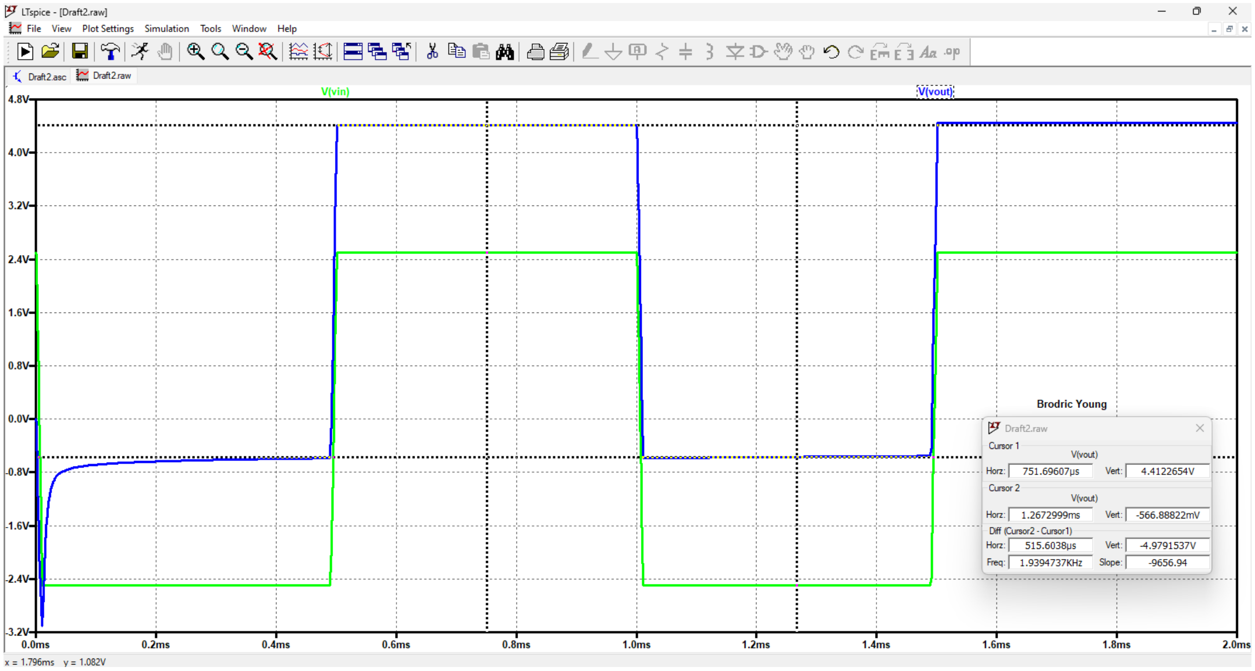

Figure 4: Input and Output Waveforms of the Positive Clamper Circuit used for **Part 2** of the Clipper, and Clamper Lab.

<u>**Discussion and Conclusions Questions:**</u> (For the following questions use your own words along with complete sentences. Points are to be deducted for AI generated answers.)

1. Explain in detail how the parallel clipper circuit of **Figure 1** operates through a full positive cycle of the 10 V peak input sine wave, aided by the graph of diode current in **Figure 2**. In your explanation include descriptions of each of the following diode operating states: D2 not sufficiently reverse biased for Zener breakdown, and D1 forward biased and D2 operating in Zener breakdown. (6 points.)

   When we're at the 10V peak input, we can see from the graph that the voltage across the two diodes (Vout) is about 6.9V. This is because D2 will be in reverse breakdown and will do all it can to hold the voltage at 6.2V across it plus D1 will be forward biased and will drop the 0.7V knee voltage across it which adds to the 6.2V of D2 to give a total of 6.9V across them both. The remaining 3.1V will be dropped across R1.

2. Explain in detail how the positive clamper circuit in **Figure 3** operates, starting with the negative portion of the input waveform, explaining the resulting capacitor voltage and output voltage, followed by the same information for the positive portion. In your explanation assume an AC input waveform varying from $-V_p$ to V_p, i.e., having a $2V_p$ peak-to-peak value. Also use the simplified diode model of 0.7 V for a forward biased silicon pn junction diode for arriving at capacitor and output voltage values. And finally

mention the peak-to-peak output voltage of the clamper circuit as the difference between the maximum and minimum output voltages. (6 points.)

On the negative portion, with our input of -2.5V, D1 would be forward biased and there will be 0.7V across it so Vout would have to be -0.7V. This gives a voltage of 1.8V across the capacitor. On the positive portion, with our input of 2.5V, D1 would be reverse biased and acting as an open switch. So Vout will be the input voltage plus the capacitors voltage which is 2.5V plus 1.8V resulting in 4.3V. The peak-to-peak voltage would be the max minus the min which is 4.3V minus -0.7V which gives a 5V peak-to-peak as we would expect from a 2.5V peak sinusoidal source. The DC voltage from it has just been shifted up.

<u>LTspice Clipper and Clamper Lab Grading Rubric:</u> **This is a CAD lab to be done individually, rather than in teams, although please help each other out if/when opportunities arise, avoiding plagiarism. Submit an electronic version of a lab report to receive credit for doing this lab.** The goal of your **lab report is to provide sufficient documentation so that the lab can be repeated if necessary.** Therefore, simply add to this document to arrive at your lab report, as all of the explanatory text, procedures and Discussion and Conclusion questions contained in this document are required for a complete lab report. So for your lab report, **add a cover page, your results, along with your answers to the Discussion and Conclusions questions to the existing lab document.** Your answers to the **Discussion and Conclusions** questions are to **be uniquely yours** and not a copy of someone else's answers to these questions. Your cover page is to include class, lab title, and author. The rubric below does not need to be included in your lab report.

Lab Report Item	Points
Cover Page	2
Part 1 – Parallel Clipper Circuit **Figure 1** schematic. (9 points total. 3 points for correctly connected schematic, 1 point for voltage source with **SINE(0 10V 1kHz)** configuration statement, 1 point for **.tran 3ms** simulation command, 2 point for zener diode symbols and D1N5234 diode type for D1 and D2, 1 point for D1N5234 model statement included, 1 point for name included on schematic.) **Figure 2** Screen Capture. 9 points total, 1 point for left vertical axis limits of 12 V to -12 V, 1 point each for correct looking **V(vin)**, **V(vout)** and **I(R1)** or **-I(R1)** waveforms (3 points), 2 points for including cursor pane on the plot pane, 2 points for minimum and maximum **V(vout)** magnitude values in cursor pane of approximately 6.9 V (-0.5 points each for value < 6 V or > 8 V), 1 point for including name as annotation.	18
Part 2 – Positive Clamper Circuit **Figure 3** schematic. (7 points total. 3 points for correctly connected schematic, 1 point for voltage source with **PULSE(-2.5 2.5 1us 10us 10us 480us 1ms 20)** configuration statement, 1 point for **.tran 2ms**	14

simulation command, 1 point for 1N4148 diode type for D1, 1 point for name included on schematic.) **Figure 4** Screen Capture. (7 points total. 1 point each for correct looking **V(vin)**, and **V(vout)** waveforms (2 points), 2 points for including cursor pane on the plot pane, 1 point for minimum **V(vout)** value in cursor pane of approximately 0 V (-0.5 points for minimum values $<$ 0.1 V or $>$ 0.1 V), 1 point for maximum **V(vout)** value in cursor pane of approximately 5 V (-0.5 points for maximum values $<$ 4.7 V or $>$ 5.3 V), 1 point for including name as annotation.)	
Discussion and Conclusions	12
Grammar and Professionalism	4
Total	50

Please give feedback on errors you find in this document.